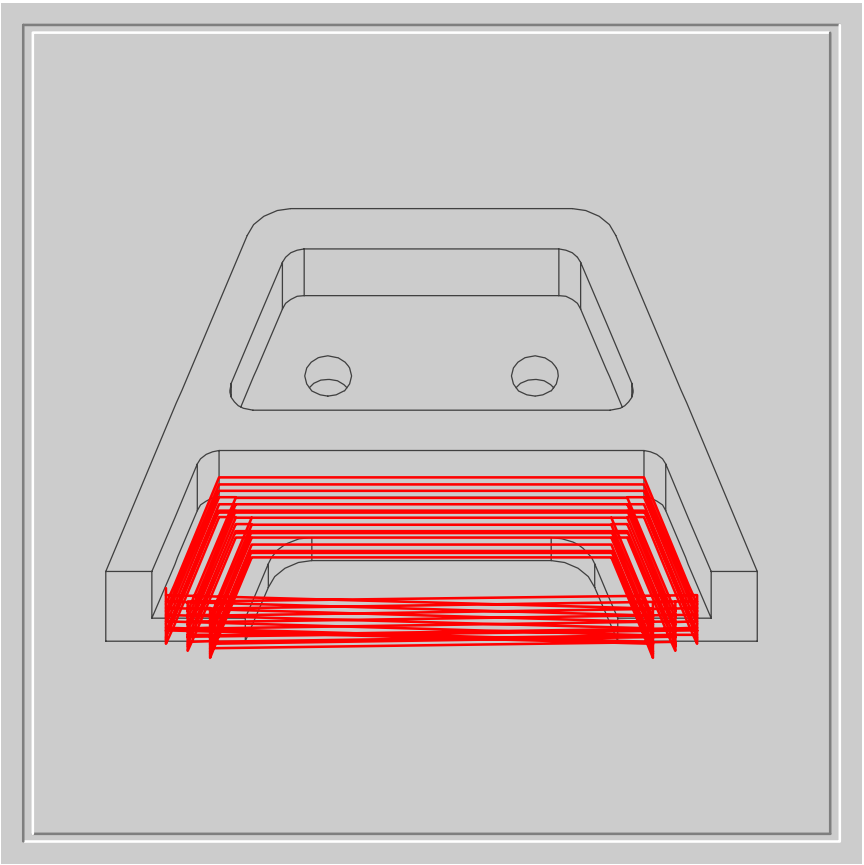


Creating Profile Operations

I-DEAS™ Tutorials: Milling Projects

Profile is a 2 1/2-axis mill operation typically used to create swarf or surface finish passes. It's sometimes known as waterline machining.

In this tutorial, you'll learn how to define axial depths and side passes for profile operations.

Learn how to:

- generate axial depths and side passes

Before you begin...

Prerequisite tutorials:

- all tutorials under the Modeling Fundamentals menu
- Introduction to Generative Machining
- Building a Setup Assembly
- Generating In-process Stock and Checking Validity
- Working with Tools and Tool Catalogs
- Picking Holes
- Setting Machining Parameters for Hole-making Operations
- Creating Face Mill and Volume Clear Operations

Retrieve the model file that you created in the Creating Face Mill and Volume Clear Operations tutorial.

File
Open

Model File Name: volume
OK

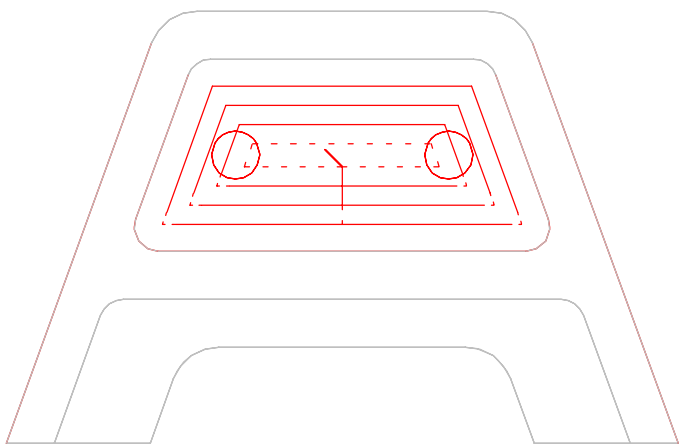
Make sure you're in the following application and task:

Application: Manufacturing
Task: Generative Machining

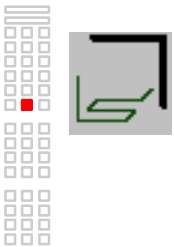
Set your units to Inch (pound f)

Options,
Units

Inch (pound f)



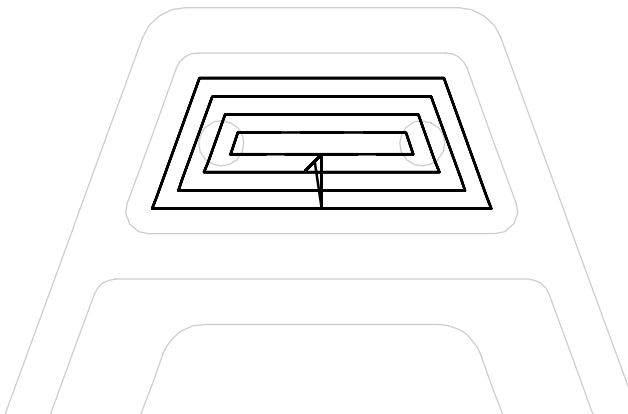
Change the display of slow motions from the previous tutorial.



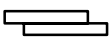
Toolpath Display Options form


Slow: ———


OK



Recovery Point



File
Save

Warning!
If you're prompted by I-DEAS to save your model file, respond:

No

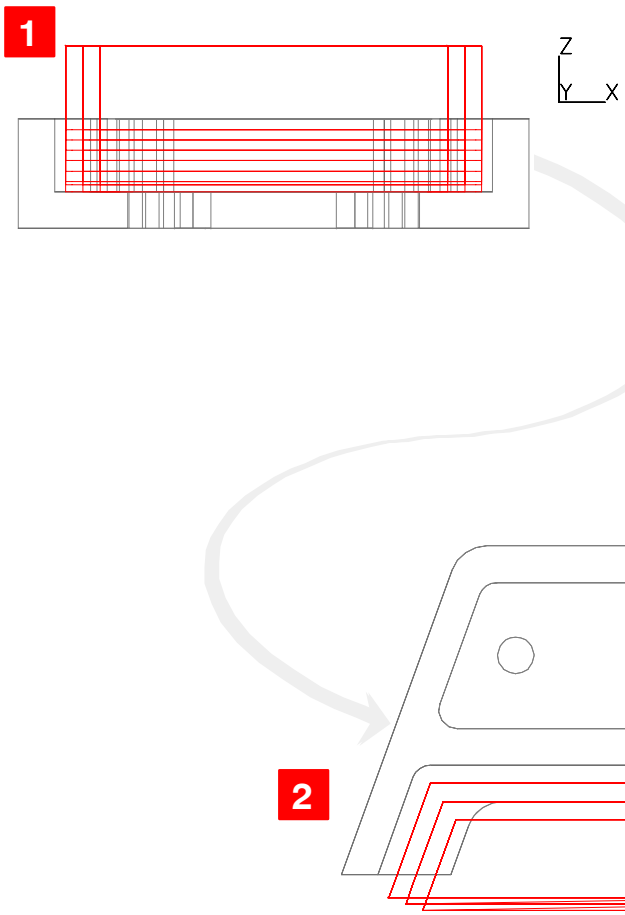
Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

If you make a mistake at any time between saves and cannot recover, reopen your model file to the last save and start over from that point.

Hint
To reopen your model file to the previous save, press Control-Z.

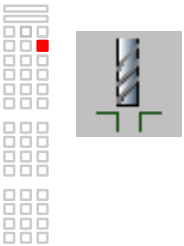
In the next steps, you'll create a profile operation. You'll define the number of depths to ensure the pocket is machined completely. Then, you'll specify the number of side passes to machine the bottom surface of the cavity.

1. axial depths
2. side passes



What: Create a profile operation.

How:



Operation Selection form

Category: Milling

Type: Profile

Create

Operation Specifcation form

Name: Open Profile

Don’t close the Operation Specification form.

What: Pick the surfaces to be machined.

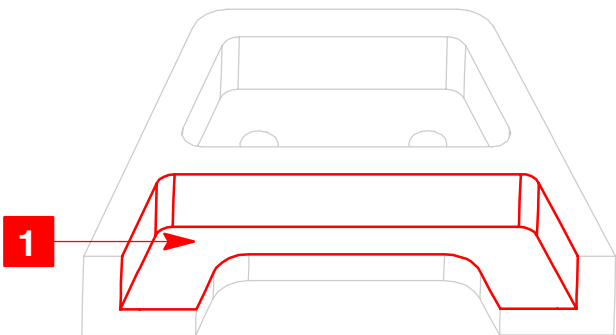
How: Remember to hold the Shift key as you pick the six surfaces composing the cavity.

Operation Specification form

Surface Selection form

 *Deselect All*

1 Double–click on F1 to select the surfaces.



Surface Selection form

 *Dismiss*

Don’t close the Operation Specification form.

What: Use a 1/2” diameter end mill to machine the surfaces.

How: You’ll open the project supplied with the software for this example.

Operation Specification form

Cutting Tool Specification—Mill form

Find...

Item Selection form

1/2” dia End Mill

OK

Cutting Tool Specification—Mill form

OK

Don’t close the Operation Specification form.

What: Enter the maximum and minimum depths to define how deeply the tool can cut. Remember that the maximum and minimum values represent a percentage of the tool diameter based on the modifier menu next to the fields. The software then calculates the number of depths based on the size of the pocket and these values.

How:

Operation Specification form

Machining Parameters: Cut form

Axial Depths...

Axial Depths form

Maximum Depth Of Cut: 30

Minimum Depth Of Cut: 1

Generate Depths

Things to notice

The generated depths never violate your maximum and minimum values. You can verify the size of the depths by multiplying these values by the tool diameter:

- 30% (maximum depth) x .5” (tool diameter) = .15
- 1% (minimum depth) x .5” = .02

Each depth of cut must be between .15 and .02 inches from the previous depth.

Don’t close the Axial Depths form.

What: Add a new depth. Although the depths are generated by the software, you can modify them or enter them all manually.

How:

Axial Depths form


–1.00000 Critical Depth


–0.9 (in the field under the table)

Things to notice

The software inserts the depth before the one you highlighted, regardless of its value. However, in this step, you insert a depth that follows the descending order of the cuts.


OK


Don’t close the Machining Parameters form.

What: Add two side passes. The software automatically makes a finish pass along the selected surfaces. You add two side passes to ensure the horizontal surface is completely machined.

How: Enter two side passes in the first field. Then enter a stepover of .25 in the second field.

Machining Parameters: Cut form

Side Passes...

Side Passes form

2 (first field)

0.25 (second field)

Things to notice

The table displays three columns:

- the number of side passes: 2
- the stepover: 0.25
- the total distance between the side pass and the part: 0.5

OK

Don’t close the Machining Parameters form.

What: Specify in-plane entries and exits. You use in-plane entries and exits in this instance because the selected surfaces represent an open volume. The end mill can plunge outside of the part to the desired depth without touching material, then the side of the tool can engage the stock.

How:

Machining Parameters: Cut form


Cut...
Entry...

Machining Parameters: Entry form


Entry Type: In Plane


Angle: 0


Entry...
Exit...

Machining Parameters: Exit form


Exit Type: In Plane


Angle: 0

Things to notice

As you enter the parameters, the software displays a preview of the entry and exit on the part.

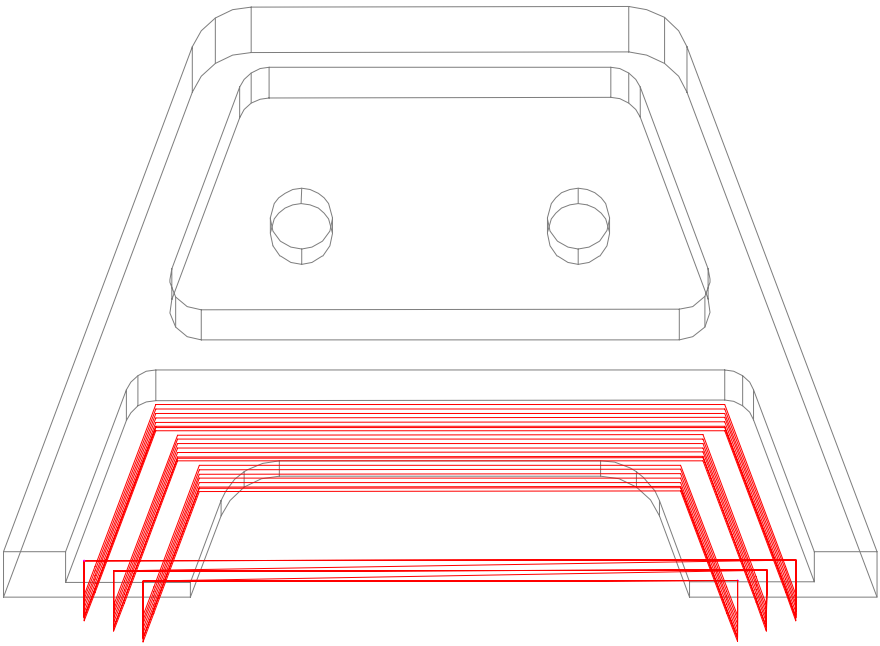

OK


Don’t close the Operation Specification form.

What: Generate the toolpath.

How:

Operation Specification form



Things to notice

The toolpath contains the number of generated depths plus the one you added. The software also has three side passes—the one the software automatically created plus the two you added. Notice the in-plane entries and exits outside of the part.

Recovery Point

File
Save

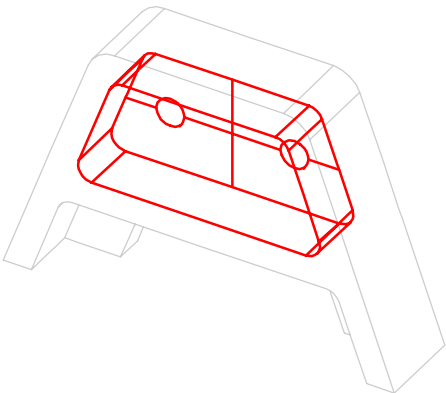
On your own...

What: Create another profile operation to finish the cavity that you roughed out in the previous tutorial.

How: Use the following parameters:

Name the operation: Closed Profile

Select surfaces



Use the same 1/2” dia End Mill tool

Specify the cut parameters

 *Constant Step*

 *Constant Step*: 30

Cut parameters continued on the next page.

On your own...

Delete the side passes inherited from the Open Profile operation.

Side Passes...

2 0.25000 0.50000

OK

Specify the entry parameters

Cut...

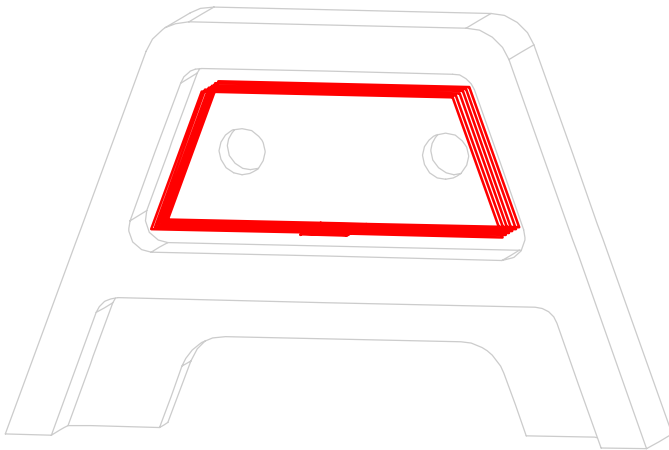
Entry...

Start Near: 0, 0, 5

OK

On your own...

Generate the toolpath



Tutorial wrap-up

You've completed the Creating Profile Operations tutorial.